MODELING AND SIMULATION SYSTEMS FOR 21 CENTURY

Antal Csuka

1. SUMMARY

The ANSYS program is a commercial framework primarily targeted at mechanical simulation research and industry. The (FEM) Finite-Element Method developed initially in mechanical and civil engineering today is commonly used in mechanics, thermodynamics, electromagnetic analysis and electronic appliance development. The method is of increasing importance in other fields such as fluid mechanics and fluid dynamics. Linear, transient and nonlinear problems in fluid flow, electromagnetic wave propagations, energy transfer, may be solved relatively easy using ANSYS program tools. This paper discusses briefly the evolution of methods and engineering systems used to analyse helicopter flight dynamic.

2. INTRODUCTION

The application of the methods of Computer Aided Engineering (CAE) in industrial design is looking back for about a fifty years old history. CATIA V4/V5 is the one of the first CAD program to provide 3D modelling, developed by the French aerospace company Dassault Systems. Most of engineering simulator tools, allow the user to analyze hypothetical situations and response of complex systems in various conditions. Features, such CAD associativity and an intuitive workflow environment lead to reduce the time needed to solve complex engineering problems. Because of the difficulty of modelling application behaviour in detail, most of earlier simulation tools had simple application models to build complex systems. Today simulation programs, such ANSYS drawing tool, have a simple but complete CAD software tool to create mechanical drawings in a quick and easy mouse driven Windows environment. ANSYS is one of the primary analysis tool for advanced simulations with complete built–in CAD modelling system that uses the FEM for solving integral and partial differential equations. An analysis of fluid mechanics is now unthinkable without the finite element method. The paper is organized as follow: In next section FEM short history will be presented. In section 3 due to atmospheric turbulence a helicopter flight problems will be described. In section 4 the ANSYS CFX /CFD features will be briefly described.

3. FEM HYSTORY

The FEM has been in existence for more than 60 years. One of the authors, John Argyris invented this technique in World War II in the course of the check on the analysis of the swept back wing of the twin engine Meteor Jet Fighter. He was responsible for the original formulation of the matrix force and displacement methods, the forerunners of the FEM. It was wartime that necessitated the sudden explosion of knowledge and methods such aeronautical structures analysis of complex structural systems. In 1938 Professor Dr. Arthur C. Ruge discovered the bonded wire strain gauge (gage) and perfected it in 1939. During World War II strain gauge measurement were widely used for structural testing in the aircraft industry. Practically all aircraft calculations were performed by some kind of force method in which forces and stresses appeared as unknowns. In those time mostly transducer based measurement and methods were used to obtain results. In November 1943 the American NACA director of Aeronautical Research in his letter to Professor Ruge concluded: “There should be no doubt that each of these apparently insignificant boundless of wire and scraps of paper is contributing much to the success of thousand of our military aeroplanes”(Ref.1). In 1944 towards the end of World War II with the advent of jet propulsion, arose the necessity of developing high subsonic speed fighters and fighter bombers. The Meteor fighter designed by Glouchester Aircraft was the answer to the German ME-262 that suffered many failures due to the poor structural design of its swept back wings. In 1944 John Argyris was working with HL Cox at the National Physical Laboratory at Teddington, London. During course of that work Argyris realized that the force method was not suitable for this problem due to the great difficulty in developing the self equilibrating systems. He toyed with the idea of the displacement method and it suddenly occurred to him that the triangle was incredibly well suited to this odd swept back wing structures. Using a number of triangles it was possible the discretization of the high speed subsonic structures (Ref.2). Elementary matrix code was designed and with this new method the wing structure was analyzed. Simulation programs such ANSYS today uses more sophisticated methods to create mesh structures and elements (Ref.3). The basic concept of the FEM is that although the behaviour of a function may be complex when viewed over the large region, a simple approximation may suffice for a small sub-region. The total region is divided into a number of on overlapping sub-regions called finite elements. In two dimensions FEM uses simple polygons such triangles and squares, in three dimensions tetrahedral, hexahedral, prism or pyramids (Ref.2),(Ref.5). Sometimes for more accurate analysis a combination of this figure is useful. The accuracy of the results, depend upon the number of elements in the model. With continuous grown in computational capacity also new mathematical procedures developed in mesh generation allow for users to build larger and larger models. Today, large models consist of several million elements and can be solved in relatively short time (Ref.4). Simulation technology today enables engineers to explore alternatives trying different ideas, to see that their ideas work, or doesn't work, to be their own innovator.

4. ANSYS WORKBENCH-CFX/CFD

Fluid simulation (CFX) and fluid dynamics (CFD) computational module integrated into ANSYS Workbench is an efficient engineering method for simulating the behaviour of systems. Both computational systems allows for user an in depth analysis of fluid mechanics in various type of physical processes. CFX is one of four ANSYS software tools designed especially for understanding the motion of liquids and gases. ANSYS CFX provides a wealth of data that can't be extrapolated from bench tests (Ref.6). Vector, streamline and contour plots give rapid insight into the three-dimensional nature of the flow. Easily-generated animated movies help to effectively communicate these results. This application is crucial in many branches of engineering. CFD is an efficient engineering method to calculate flow fields distribution in detail used from large scale of applications (aircraft turbine) to nanotechnology (MEMS) due to the scalable nature of fluid dynamics. ANSYS Workbench is the common graphical user interface based on implementation of FEM and computational methods in fluid dynamics which support parametric variations, including CAD geometry boundary conditions, material properties. The ANSYS Workbench platform automatically forms a connection to share the geometry for both the fluid and structural analyses, minimizing data storage and making it easy to study the effects of geometry changes on both analyses. The toolbox contains the systems to build the project and provides a single environment for simulation from start to finish. Geometry can be created directly in ANSYS using proper CAD design tool using key point, lines, arcs, or use imported geometry file from solid modelling CAD systems such CATIA, Pro-Engineer. CFX-Flow and CFD software has direct access to CAD through the geometry interfaces of ANSYS Workbench. The IGES (Initial Graphics Exchange Specifications) neutral file is a common format used to exchange geometry between computer programs. The current version of ANSYS software products is 13.1 (Ref.7).

5. HELICOPTER RESPONSE TO ATHMOSPHERIC TURBULENCE

The mathematical modelling of helicopter dynamics is very difficult due to dynamic behaviour. The airflow over the rotary wing can never be characterised by linear aerodynamics. Nonlinear phenomena of flow problems caused by turbulence are often difficult to interpret because of their complexity. The main problem is that the behaviour of helicopter in flight is a combination of large number of subsystems. The subsystem components are; fuselage, empennage, main and tail rotor elements, power-plant (Ref.9). The individual interaction between subsystems may be described by well known linear equations such as Newton's motion laws, conservation of energy and so on (Ref.8). One of the main goals of mathematical analysis of systems, are the investigation and description of system behaviour in case of different environmental condition. The helicopter response to an atmospheric disturbance can be measured in terms of the force and moment derivatives. The analysis must be extended to the modelling of atmospheric disturbance because the main rotor is a dominant lifting component on helicopter. During flight, the angle of attack of rotating blade is usually between 0,5 to 8 degree (Ref.9) (Ref.10).The statistical discrete gust approach to turbulence modelling used for airplane fixed wing applications appear to be ideally suited for "low level" helicopter applications such helicopter modelling. The extension of fixed-wing methods to helicopters are generally applicable at operating heights above about 60m, to helicopter models as the BELL

TH-67 model of scale 8:1 above 2m (Figure 1.). The most common form of turbulence model involves the decomposition of the velocity into frequency components. The (PSP) Power Spectral Density method contains information about the excitation energy within the atmosphere as a function of frequency is described by von Karman PSD function. The function is detailed described in literature (Ref.11). Several studies have concentrated on fatigue analysis and hub vibratory loading problems, investigated factors that alleviate the fuselage response. Rotor-fuselage penetration effects tend to dominate the alleviation with secondary effects due to rotor dynamics and blade elasticity (Ref.15).

The main goal of flow analysis using the ANSYS CFX simulation program are the determination of location of extreme value of pressure and air velocity around the fuselage. The helicopter in flight is subject to three kinds of efforts: its weight, the aerodynamic and the propulsion effort. The aerodynamic and propulsion effort depend upon the characteristics of the atmosphere, interaction of helicopter body and rotating wing upon the drag co-efficient of this parts. The atmosphere is characterized by the state of the air; temperature, pressure, humidity and its velocity. (Ref.12) (Ref.13).The interference phenomena between horizontal stabilizer and main rotor wake, is a subject of many study because of increasingly importance in design of rotating wing and fuselage. The lift and drag are both proportional to the area of the wing and depend upon the cross section type of rotating wing. The applied wing sections type to simulate the interaction between rotor blade and empennage are the NACA 0012 section type, as can see on Figure 1. (Ref.16).

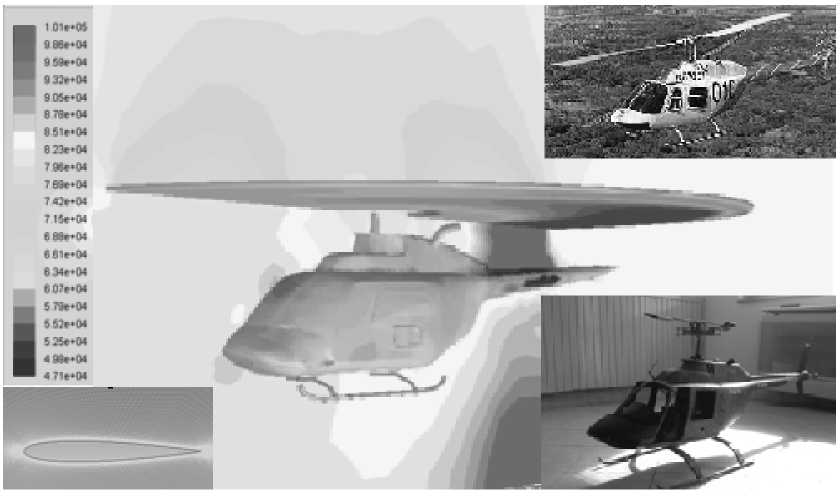

Figure 1.
Pressure distribution results on Bell TH-67 "Creek" helicopter model

Table 1.
Helicopter model specifications

Parameter	**BELL TH-67 "Creek" 1993 – USA**
Scale	8:1
Fuselage length	1,4 m
Fuselage width	0,21 m
Fuselage height	0,57 m
Main rotor diameter (proposed data for simulation)	1-1,5 m
Tail rotor diameter (proposed data for simulation)	0,17-0.25
Rotating wing section type	NACA-0012
Un-instrumented empty weight (without gear engine, motor, fuel, onboard electronic system, rotor blade as can see on Figure 2 –right low corner)	2,73kg

The NACA airfoil type designed for comparatively low speed applications seems to be suitable for helicopter main rotor systems. This shape is a symmetrical airfoil type of 9% thickness (Ref.14). The simulation results obtained on helicopter model presented is capable for predicting the best vertical and horizontal location of the stabilizer. The model presented on Figure 1. shows the velocity of air flow and pressure distribution around the fuselage.

6. CONCLUSION

Finite Element Analysis and simulation technique has eliminated most of physical prototype stages. Using ANSYS we can discover how to best utilize simulation techniques in the prototype development process saving time and money. Using this technique it is possible to reduce the need for expensive prototypes because it provides comprehensive data that is not easily obtainable from experimental tests. The ANSYS simulation method has been found to be effective in predicting pressure distribution on fuselage. By the analysis of the pressure distribution and flow results shown in Figure 1. it can be concluded that the airflow causes unequal distribution of forces that must be take in account by design of empennage. The horizontal stabilizer will be mounted on lowest pressure region of tail.

7. LIST OF REFERENCES

[1] Peter, K. Stein: Strain gage history and the end of the twentieth century, Experimental techniques, March/ April 2001.

[2] Kythe, P. K, and Wei, D.: An introduction to linear and nonlinear finite element analysis, Birkhäuser, Boston, 2004.

[3] ANSYS Conference Proceedings, Pittsburgh, Pennsylvania, April 17-20, 1983.

[4] Gockenbach, Mark S.: “Understanding and implementing the finite element method”, Society for Industrial and Applied Mathematics, Philadelphia, 2006,

[5] Müller, Gerhard: Finite Elemente, Hütig, Heidelberg, 1989,

[6] ANSY CFX tutorial; http://seam.ustb.edu.cn/UploadFile/20080512040742250.pdf, (13.01.2011),

[7] ANSYS 13.1, http://www.ansys.com/, http://www.econengineering.com/ (21.01.2011),

[8] Bramwell, A.R.S.: Helicopter dynamics, Arnold Publishers, Avon,1976,

[9] Locke, James E: “Balanced continuous turbulence gust loads using solution”, http://www.mscsoftware.com/support/library/conf/auc99/p02099.pdf (01.13.2011),

[10] Dynamic Analysis of geared Rotors by finite Elements”, National Aeronautics and Space Administration, Cleveland, Oh. Levis Research Center, 1990,

[11] John, Seddon: Basic helicopter aerodynamics, PSP Proffesional Books, Oxford, 1990,

[12] Brown, Eric M.: The helicopter in civil operations, Granada, London, 1981,

[13] Jean-Luc Boiffier: The Dynamics of Flight-The Equations, John Willey and Sons, New York, 2000,

[14] Leishman, J. Gordon: Principles of helicopter aerodynamics, http://books.google.hu/, (22.02.2011.),

[15] Timothy M. F. and Richard, E. B.: “Main rotor–Empennage Interaction and its Effects on Helicopter Flight Dynamics”, http://www.legacy.aero.gla.ac.uk/Research/rotorcraft/rotor/publications/pubs/AHSForum07_TMF_REB.pdf, (24.02.2011.),